Via Wizard 2.0
User Manual

Current as of 1/7/06

1. Introduction

The Via Wizard has been developed to simplify project creation in HFSS v10. Using a straight-forward GUI, a user can enter all parameters necessary to create an arbitrary via array. Most projects will be launched ready to solve in HFSS however the user may easily modify or add to the geometry created by the wizard. Many variable parameters have been added to the project so the user can easily perform parametric analysis.

Although every effort has been made to ensure accurate projects, the user should verify everything is setup as intended. The Via Wizard is available free of change and may be distributed without a license. Updates will be made available as appropriate and may be downloaded from 3DViaDesign.com or Ansoft.com/ots.

1.a Quickstart Instructions

1. Launch "Via Wizard GUI.exe."
2. Fill in the desired information for each of the tabs: Stackup, Padstack, Via.
3. Click "Generate Project."
4. Typical Via projects are now ready to solve.

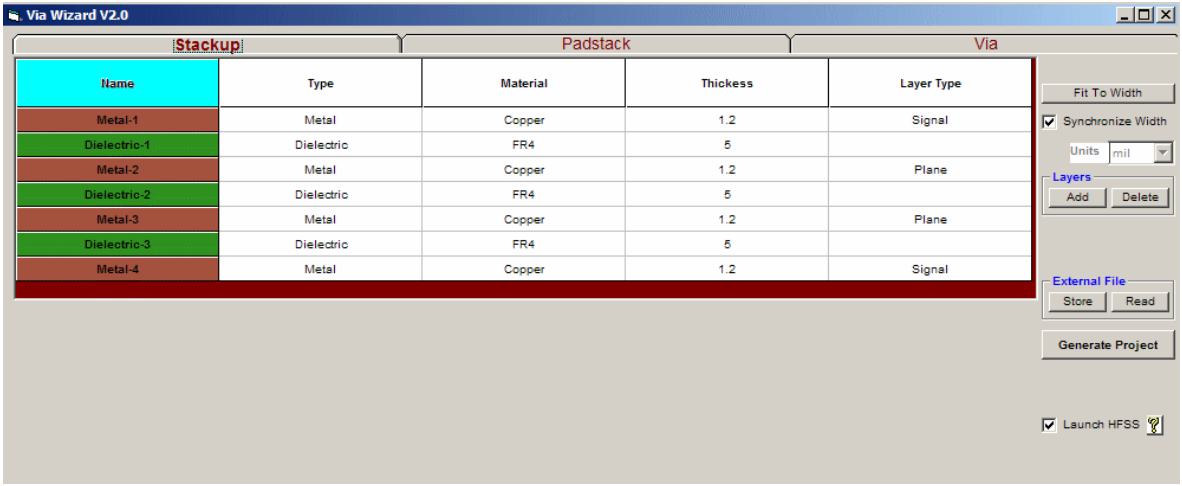
1.b Basic Program Operation

The Via Wizard consists of two parts: The "Via Wizard GUI.exe" and "HFSS_via_wizard.exe." The GUI will accept user inputs that describe everything necessary to create an arbitrary via array. After all of the information has been entered, the GUI will generate a text file called "TempFile.txt" that is input into HFSS_via_wizard.exe." This program will then parse the data and generate a VBscript called "HFSS.vbs" that can be read directly into HFSS. Please refer to the "TempFile_specs.xls" for information on the temp file format.

2. GUI Parameter Description

There are three tabs for the GUI used to describe all parameters necessary for the via array. Each of the input parameters is described below:

2.a Stackup



The default project contains 4 Layers. Additional layers can be added to the bottom of the stackup using the **ADD** button on the right. Bottom layers can be removed using the **Delete** button on the right.

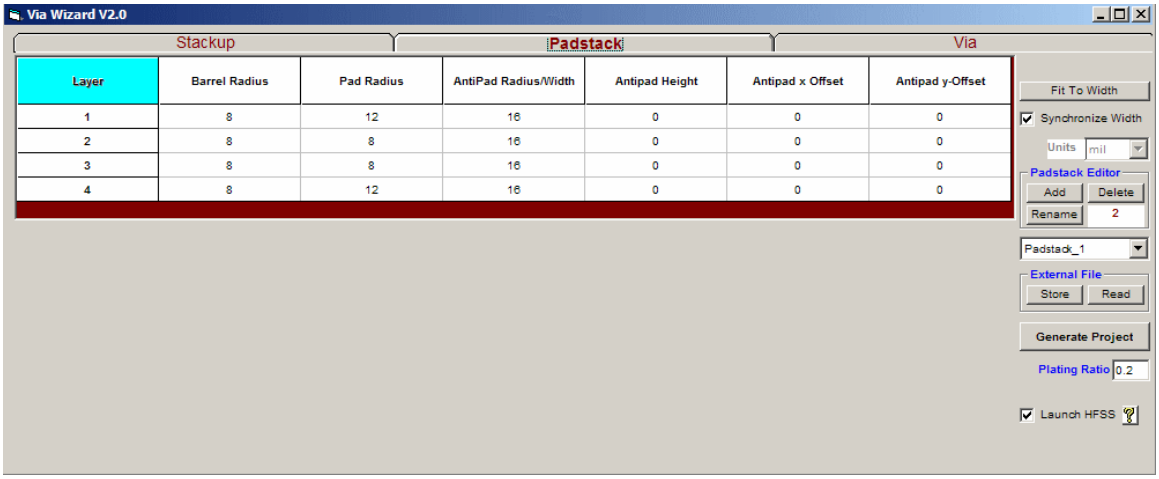
Type describes the layer type for the stackup. The stackup always alternates between Metal and Dielectric. The top and bottom layers are always Metal

Material describes the material properties that will be used in HFSS. By default Copper is always used for Metal layers and FR4 is always used for Dielectric layers. Future versions may contain additional material selections. To use different materials, change the material type in the created HFSS project.

Thickness describes the layer thickness in mils. The default stackup uses 1-oz copper with 5 mil dielectric.

Layer Type describes which layers are signals and which layers contain planes. To change the type, select it from the pull down menu. A valid project must contain at least one plane.

2.b Padstacks



The padstack display changes based on the number of layers defined. For each Metal layer that is defined in the Stackup tab, a row is inserted for the vias. Each of the rows may be defined independent of each other. By default, two padstacks have been defined. Choose the padstack you would like to edit using the *Padstack Pull Down Menu* on the right. Padstacks can be renamed by using the **Rename** button on the right. Padstacks may be added or deleted using the **Add** and **Delete** buttons on the right. In the lower right corner you can define the **Plating Ratio** for each padstack. This is the ratio of Plating Metal to Drill Size. The default value is 0.2 and must be between 0 and 1.

Layer describes the metal layer that is described by the padstack.

Barrel Radius describes the radius of the via drill hole. This is maximum radius of the plated through hole.

Pad Radius describes the pads on each layer. By default the radius is set equal to the Barrel Radius indicating no pad exists on the internal layers. On the external layers the default pad size is set to 12 mils.

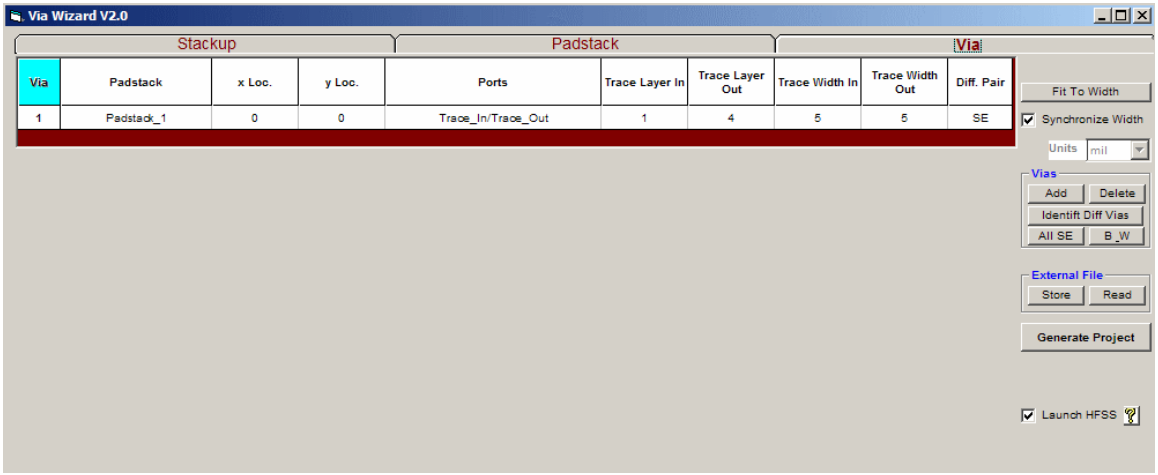
Antipad Radius/Width describes either the radius of circular antipads or the width of rectangular antipads. Circular antipads are drawn if Antipad Height is 0, otherwise a rectangular antipad is drawn. By default circular antipads are defined that are 16 mils.

Antipad Height describes the height of rectangular antipads. This value should be set to 0 to describe circular antipads.

Antipad X Offset describes the x offset from the center of the via barrel. This offset is applied to the center of circular antipads or the middle of the rectangular antipad's width. The default for this value is 0, meaning the antipad is perfectly centered about the via barrel.

Antipad Y Offset describes the y offset from the center of the via barrel. This offset is applied to the center of circular antipads or the middle of the rectangular antipad's height.

2.c Vias



The Vias tab is where the array of vias is defined. Any number of vias can be added to the project by clicking the **Add** button on the right. You may delete the last via added by using the **Delete** button. If you like to delete a via other than the last one added, select the via by clicking on the number in the leftmost column and click **Delete**. If some vias have been paired together as a differential pair, you can color code them by clicking **Identify Diff Vias**. If you would like to remove the color coding, select **B_W**. You can convert all differential vias to single ended by clicking **All SE**. You must convert all vias to single ended in order to delete a via from the array.

Via describes the number of the via.

Padstack is a list of all available padstacks as previously defined. Select the desired padstack from the pull down menu.

X Loc is the x location of the via center on a Cartesian plane.

Y Loc is the x location of the via center on a Cartesian plane.

Ports is a pull down menu which allows you select the type of port associated with that via. Choices are *None*, *Trace in-Trace out*, and *Coax in-Trace out*.

None only places a via and does not attach a port. Use this selection for Ground vias or unconnected signal vias.

Trace in-Trace out draws two traces that connect to the via. Traces are always drawn in the +/- X direction. The trace into the via goes from the highest X extents of the project to the via. The trace out of the via goes from the via to the lowest X extents of the project. Wave ports are used as excitations.

Coax in-Trace out draws a coax launch from the top of the project into the via. This is similar to a press-fit connector. A trace is also drawn out of the via escaping to the lowest X extents of the project. Wave ports are used for excitations.

Trace In Layer defines the layer the trace is drawn coming into the via. Only signal layers are displayed in the pull down menu.

Trace Out Layer defines the layer the trace is drawn escaping from via. Only signal layers are displayed in the pull down menu.

Trace In Width defines the nominal width of the trace into the via. Traces are drawn as trapezoids to capture over etching. The value defined here is used as the base of the trapezoid with the top oriented upward.

Trace Out Width defines the nominal width of the trace escaping the via. Traces are drawn as trapezoids to capture over etching. The value defined here is used as the base of the trapezoid with the top oriented downward.

Diff Pair defines which vias are paired together. Pairing vias together will draw an antipad that surrounds both vias and also link the escape routing together. For more details, please refer to **Differential Vias** in the *Via Designs* section.

2.d Buttons Common to All Tabs

Some buttons are available for use on each of the tabs.

Fit to Width scales the columns to match the width of the GUI window.

Synchronize Width scales the width of the columns are the window is resized.

Units is not yet active but will be in the next release. To use different units enter them into the wizard as you like and simply change them in HFSS after the project has been created with *3D Modeler->Units*. Do not check the rescale button.

Store will save the current configuration as a text file. Data from all three tabs will be saved.

Read will load a saved Via Wizard project. Data from all three tabs will be restored.

Generate Project will create the “HFSS.vbs” file that can be imported into HFSS. The file is saved in the Via Wizard project directory. To load it manually, select *Tools->Run Script* in HFSS and point to this file. If the **Launch HFSS** button is checked, HFSS will be launched and the script automatically loaded.

? is a help button that directs you to this manual and also informs you of the release date.

3. Design Conventions

For simplicity and ease of use, certain assumptions were made regarding the HFSS project setup. Any type of via design may be accomplished using HFSS. The via wizard may be used to create many of these designs ready to solve, or a base project that can be modified. Understanding the assumptions the Via Wizard makes will help you use the tool most effectively.

Traces always drawn in X-direction. Ensure the path used to draw the traces is clear from other vias. The traces used for the input ports run from the highest X extents of the project to the via. The traces used for the output ports run from the via to the lowest X extents of the project.

Highest Accuracy settings are used. All geometry and settings have been tuned to give maximum accuracy. For very large projects, users may be interested in sacrificing accuracy for solve time. To decrease solve time, certain options such as “Solve Inside” may be turned off for the conductors. Planes may also be replaced by 2D sheets with finite conductivity.

Traces are drawn as trapezoids. The trapezoidal geometry is used to emulate over-etching. The base of the trapezoid is equal to width specified in the GUI. The top is equal to base – conductor thickness. Both top and bottom dimensions are parameterized. To create a square trace, which is common for microstrip, set the top equal to bottom.

Trapezoids flip orientation to mimic core and prepreg. All traces in have trapezoids oriented upwards. All traces out have trapezoids oriented downward. The top and bottom widths may be flipped based on the stackup.

Separate dielectric sheet used for each layer. All dielectric layers are independent and may be different materials. To change the material, select the layer and click 3D Modeler->Assign Material.

Default units are mils. To use different units enter the dimensions into the Via Wizard as you normally would. After the HFSS project has been created, select 3D Modeler-> Units. Chose the units and do NOT select “rescale to new units.”

Most variables are hidden. This has been done to avoid clutter. To activate parameters of interest go to HFSS->Design Properties

Dogbone parameterized. In this case the dogbone is defined as the differential antipad. The dogbone is drawn as a rectangular clearance between differential vias. The dogbone size set equal to first differential via’s antipad dimension.

Escape routing is parameterized. For all traces, three parameterized points have been chosen for the routing. Single ended vias are drawn straight in the X- direction. For differential vias, escape routing is based on antipad dimensions of layer it is escaping.

The separation between the differential traces is initially chosen as the first vias trace width. This has been parameterized as well.

Total size of PCB is 75 mils greater than via array. The edges of the project have been padded by 75 in the X and Y directions around the extents of the via array. This has been parameterized as well.

All materials are copper and FR4 by default. Change materials manually in HFSS using 3D Modeler -> Assign Material.

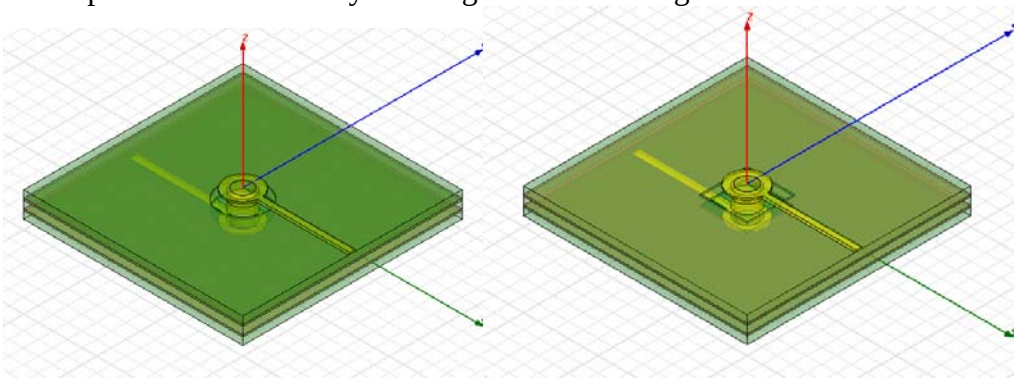
Coax launch emulates 50 ohm cable. Coax launch uses PEC center conductor set to inner barrel dimensions of via. Outer Shield is equal to 3* inner Conductor + 2mils. Dielectric is tweaked to match 50 ohm impedance.

4. Via Designs

Just about any type of via or array can be designed using the wizard. For any exotic cases, the Via Wizard may be used as a starting point. Below are a few examples of via projects and how they are created

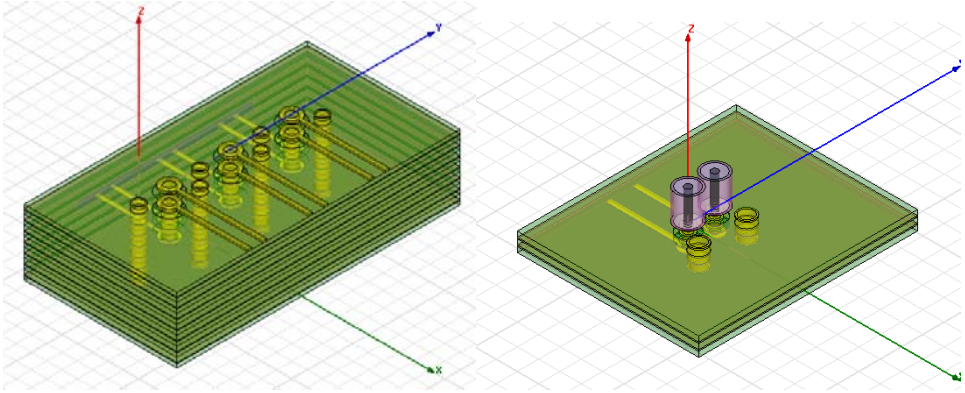
Signal Vias

-Circular antipads can be drawn by entering Radius and “0” for Antipad Height
-Square antipads can be drawn by entering Width and Height



Connector Footprint

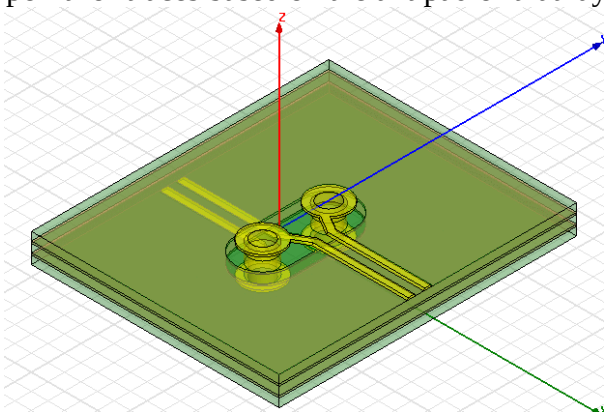
-Define pitch using X, Y locations of Ground and Signal vias
-Use Coax launch to emulate a press fit connector
-When using Coax Launch, Coax Sheath needs to be grounded by the user
-It is recommend that a PEC sheet is drawn from the Coax Sheath to the top of the nearest ground via



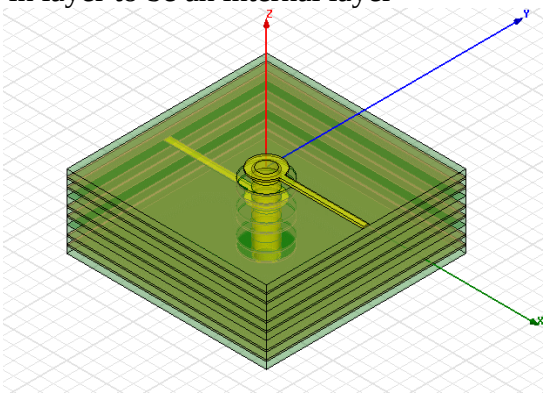
Differential Vias

-Defining Diff Pairs in the GUI will create an elliptical antipad between the two vias. The antipad dimensions will be based on the antipad radius of the via with the lowest Y location.
-Dogbone has been parameterized
-Set pad/antipad to 0 if you don’t want dogbone on layers

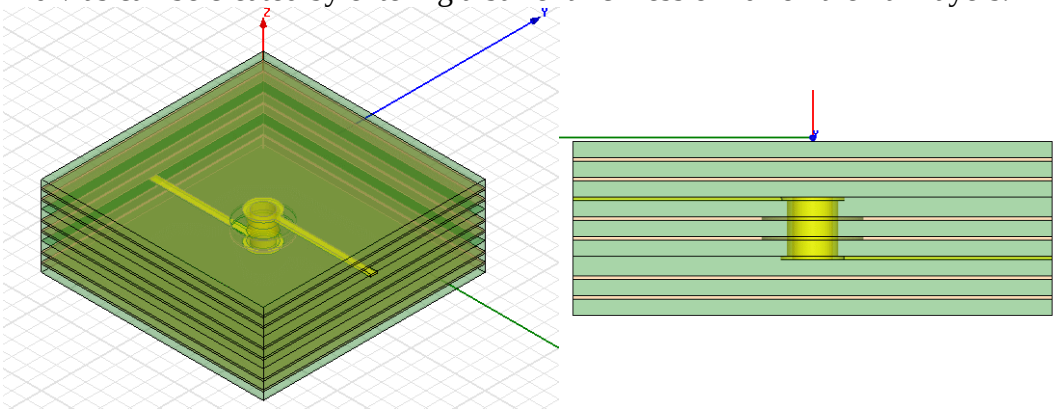
-Differential routing has been parameterized
-Trace spacing based on trace width of first via
-Location of meeting point for traces based on the antipad of that layer.



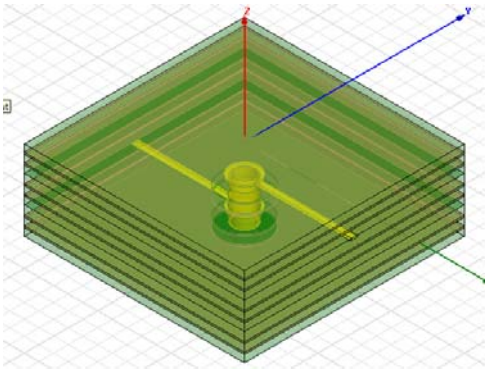
Stub Vias
-Typical PTH via
-Enter the same via Barrel radius on all layers.
-Enter trace out or trace in layer to be an internal layer

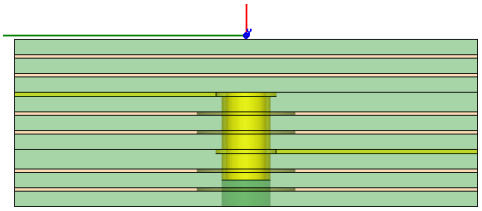


Blind Vias
-Blind vias can be created by entering a barrel thickness of “0” on the null layers.



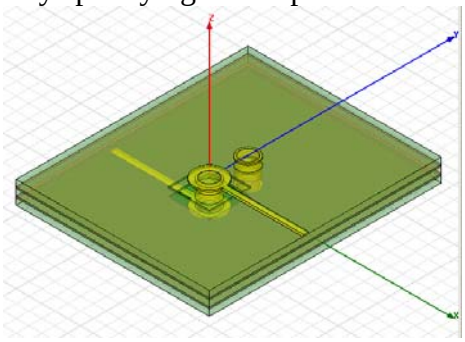
Backdrilled Vias
-Backdrilling can be investigated by changing the “Via#_backdrill” variable in HFSS





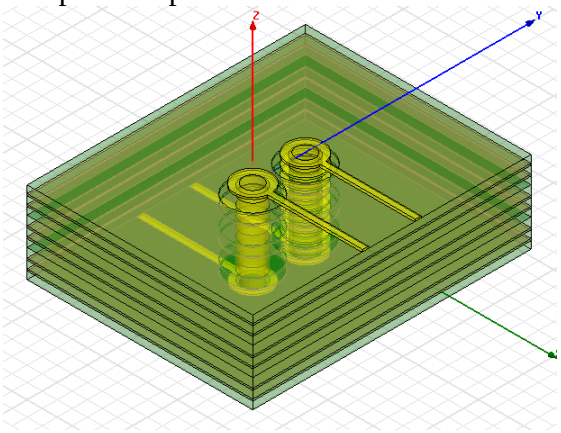
Ground Vias

-Ground vias can be created by specifying an antipad of “0” on desired plane layer



Non-Functional Pads

-To remove non-functional pads set pad radius to barrel radius



5. Advanced Usage

The Via Wizard was designed to enable creation of accurate HFSS projects with minimal user input. As a user becomes more advanced, the wizard may be used as a starting point for different types of analysis. The first section describes a few techniques to help usage. The second section lists the variables included in the HFSS project and what they stand for.

5.a Techniques

Bare PCBs can be exported with a parameterized stackup. Stackups can then be modified to include any type of geometry that is found on a PCB.

W-Elements may be exported. W-elements may be exported from the solved ports. The trace dimensions and stackup have been parameterized so different configurations may be easily tested. For best results:

-Use discrete sweep
-Use ports only solution
-Use frequency dependent materials

Lengthen Traces by deembedding. Wave ports connected to traces may be deembedded to represent long sections of traces. This is a post process operation and does not require extra solve time. As long as the trace routing is uniform, the HFSS project may be used to represent the via and trace effects.

Solve multiple variables with parametric sweeps. Any of the project variables may be swept and solved for by setting up a table. To create a table, Right-Click Optimetrics->Add->Parametrics.

Add additional variables. Geometry that does not already contain variable references may be parameterized. To add a variable to an object expand the object history and select the command you would like to parameterize. Type a variable name in place of the absolute number. Remember that only original geometry may be edited.

Turn off Autosave. Turning off autosave will help the script run faster.

Use discrete sweep to ensure passivity. Interpolating sweeps can often produce frequency data that is slightly non-passive. Non-passive models may contribute to non-convergence in time domain simulations. Discrete sweeps take longer to solve but will ensure passivity if you are having problems with transient simulation.

Use frequency dependent materials to ensure causality. Real materials have loss that increases with frequency. To produce causal models in the time domain, materials should be frequency dependent.

Purge history to reduce project size. If you command history is very long you may purge the history of some or all object by selecting them and clicking 3D Modeler->Purge History.

Delete last object in command history. To delete a specific command in the object history you must delete the previous commands in the tree.

5.b Included Variables

Via(*num*)_x
The x-location of the via center
Via(*num*)_y
The y-location of the via center
Via(*num*)_trace_in_bot
The width of the lower section of the trapezoidal trace into the via
Via(*num*)_trace_in_top
The width of the upper section of the trapezoidal trace into the via
Via(*num*)_trace_out_bot
The width of the lower section of the trapezoidal trace out of the via
Via(*num*)_trace_out_top
The width of the upper section of the trapezoidal trace out of the via
Via(*num*)_trace_in_bot
The width of the lower section of the trapezoidal trace into the via
Via(*num*)_backdrill
The depth of copper that is removed from the bottom of the via
Via(*num*)_in(*num*)_x
Location of x points for trace into the via
Via(*num*)_in(*num*)_y
Location of y points for trace into the via
Via(*num*)_out(*num*)_x
Location of x points for trace out of the via
Via(*num*)_out(*num*)_y
Location of y points for trace out of the via

For Differential Vias Only
Via(*num*)_trace_in_gap
The spacing between the differential traces into the via
Via(*num*)_trace_out_gap
The spacing between the differential traces out of the via
Via(*num*)_Layer(*num*)_dogbone
The height of the rectangular clearance between differential vias.

Hidden Variables
L(*num*)_Cond_Thickness
Conductor thickness of that layer
L(*num*)_Die_Thickness
Dielectric thickness of that layer

L(*num*)_Cond_Elevation
Elevation in Z-direction of bottom of conductor
L(*num*)_Die_Elevation
Elevation in Z-direction of bottom of dielectric
Xmin
Minimum X dimensions of project
Xlength
Length of X dimensions of project
Ymin
Minimum Y dimensions of project
Ylength
Length of Y dimensions of project
Zmin
Minimum Z dimensions of project including Airbox
Zlength
Length of Z dimensions of project including Airbox
Via(*num*)_Layer(*num*)_antipad
Antipad radius or width
Via(*num*)_Layer(*num*)_antipad_height
Antipad height
Via(*num*)_Layer(*num*)_Antipad_x_offset
Antipad registration in X direction
Via(*num*)_Layer(*num*)_Antipad_y_offset
Antipad registration in Y direction
Via(*num*)_Layer(*num*)_Antipad_xmin
Used as starting point for rectangular antipads. Computed from other variables
Via(*num*)_Layer(*num*)_Antipad_ymin
Used as starting point for rectangular antipads. Computed from other variables
Via(*num*)_Layer(*num*)_Antipad_x_center
Used as starting point for circular antipads. Computed from other variables
Via(*num*)_Layer(*num*)_Antipad_y_center
Used as starting point for circular antipads. Computed from other variables
Via(*num*)_Layer(*num*)_Antipad_x_center
Used as starting point for circular antipads. Computed from other variables
Port(*num*)_in_left_wall
Defines Y location of left edge of WavePortIn
Port(*num*)_in_right_wall
Defines Y location of right edge of WavePortIn
Port(*num*)_out_left_wall
Defines Y location of left edge of WavePortOut
Port(*num*)_out_right_wall
Defines Y location of right edge of WavePortOut

6. Known Issues

There are a few known issues with the current implementation of the Via Wizard. Most of these should be fixed in future revisions. In the meantime, some workarounds are suggested below.

Waveports not correctly defined for G-S-S-G layer configuration. The currently implementation is only valid for G-S-G stripline excitations. To fix this edit the extents of the waveport to touch the appropriate ground planes

Terminal Lines don't always move with parametric sweep. This does not typically affect the simulation however future releases will parameterize this line so it moves with the project. To fix this draw a parameterized polyline and redefine the terminal line to snap to these endpoints.

Top and bottom layers need to be signal layers. For through-hole vias, the external layers cannot be planes. It is OK to for blind vias however.

If a script fails, restart HFSS. If a VB script fails sometimes the memory is not cleared and can slow HFSS if you continue to work with the GUI.

HFSS may not be close while the Via Wizard is communicating with it. To close HFSS, first close the Via Wizard GUI.

7. Future Improvements

For the next revision of the Via Wizard, certain enhancements have already been planned. Below is a partial list or what is tentatively scheduled. Other features will be added based on user response.

Query user on signal risetime/speed. Frequency sweeps will then be added to HFSS based on the response.

More Intelligent escape routing. Traces will be able to route through a via field, not just across a line of sight path

Optimetrics automatically setup. Based on user input, optimetrics sweeps will be automatically setup.

Move pads with antipad registration. Consistent with typical PCB manufacturing process, pads should move with antipad clearances. Antipad X,Y offset will apply to both antipads and pads.

Use UDPs in HFSS to limit users to only valid geometries. This will ensure the user can only define parameters that are valid and feasible. It will also help remove clutter from the object history.

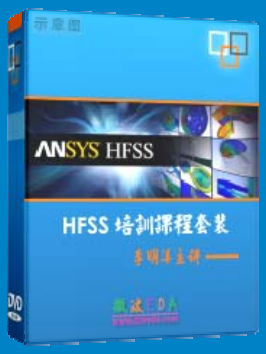
Geometry options for lower accuracy, faster solve times. In order to save time certain options can be set to aid the simulation speed. Certain things such as creating sheets for ground planes and facetizing all geometry is being considered.

Via Array Visualization. The next version will include a window so via arrays can be visualized before the HFSS project has been generated.

HFSS视频培训课程推荐

HFSS 软件是当前最流行的微波无源器件和天线设计软件，易迪拓培训(www.edatop.com)是国内最专业的微波、射频和天线设计培训机构。

为帮助工程师能够更好、更快地学习掌握 HFSS 的设计应用，易迪拓培训特邀李明洋老师主讲了多套 HFSS 视频培训课程。李明洋老师具有丰富的工程设计经验，曾编著出版了《HFSS 电磁仿真设计应用详解》、《HFSS 天线设计》等多本 HFSS 专业图书。视频课程，专家讲解，直观易学，是您学习 HFSS 的最佳选择。



HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的 HFSS 培训教程套装，可以帮助您从零开始，全面深入学习 HFSS 的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html

HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书，课程从基础讲起，内容由浅入深，理论介绍和实际操作讲解相结合，全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程，可以帮助您快速学习掌握如何使用 HFSS 设计天线，让天线设计不再难…

课程网址：http://www.edatop.com/peixun/hfss/122.html

更多 HFSS 视频培训课程：

- **两周学会 HFSS —— 中文视频培训课程**
  课程从零讲起，通过两周的课程学习，可以帮助您快速入门、自学掌握 HFSS，是 HFSS 初学者的最好课程，网址：http://www.edatop.com/peixun/hfss/1.html
- **HFSS 微波器件仿真设计实例 —— 中文视频教程**
  HFSS 进阶培训课程，通过十个 HFSS 仿真设计实例，带您更深入学习 HFSS 的实际应用，掌握 HFSS 高级设置和应用技巧，网址：http://www.edatop.com/peixun/hfss/3.html
- **HFSS 天线设计入门 —— 中文视频教程**
  HFSS 是天线设计的王者，该教程全面解析了天线的基础知识、HFSS 天线设计流程和详细操作设置，让 HFSS 天线设计不再难，网址：http://www.edatop.com/peixun/hfss/4.html
- **更多 HFSS 培训课程**，敬请浏览：http://www.edatop.com/peixun/hfss

关于易迪拓培训：

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，一直致力和专注于微波、射频、天线设计研发人才的培养；后于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计相关培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验

※ 一直专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 视频课程、既能达到现场培训的效果，又能免除您舟车劳顿的辛苦，学习工作两不误

※ 经验丰富的一线资深工程师讲授，结合实际工程案例，直观、实用、易学

联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com